Computational Fluid Dynamics Based on the Method of Space-Time Conservation Element and Solution Element

NASA Lewis Research Center
Cooperative Agreement No. NAG 3-1566

Annual Report for the Period
February 3, 1996 – February 2, 1997

Submitted by Chuen-Yen Chow, Principal Investigator
Department of Aerospace Engineering Sciences
University of Colorado
Boulder, Colorado 80309-0429

Computational Fluid Dynamics Based on the Method of Space-Time Conservation Element and Solution Element

The space-time conservation element and solution element (CE/SE) method has been further developed and applied to solve more general flow problems during the present third year of NASA support. Not only the flow problems originally proposed but also some related problems have been solved using this method. The following is a brief description of the major works that have been performed under the present support of this grant.

(1) **Numerical Simulation of an Unsteady Implosion/Explosion Process of a Polygonal Shock Wave in a Box,** *presented in the 15th International Conference on Numerical Methods in Fluid Dynamics, Monterey, CA, June, 1996.*

This problem that has been studied using a traditional TVD scheme in [1] is now solved by using a 2-D CE/SE Euler solver based on the dual-mesh pattern. In addition to the early stage of the implosion/explosion process, the later wave development which was not dealt with in [1] is also simulated in this work. The work has successfully demonstrated that i) the robustness of the present Euler scheme in handling the reflecting boundary condition for both horizontal and vertical solid walls, and ii) the feasibility of introducing the dual-mesh pattern in the CE/SE numerical scheme, which provides a strong support for the future construction of the CE/SE Navier-Stokes solver.

(2) **Numerical Simulation of Shock Reflection over a Dust Layer Model,** *submitted to the 13th AIAA Computational Fluid Dynamics Conference, Snowmass, CO, June 1997.*

Different patterns of shock wave reflection over a wedge with a smooth surface have been computed previously using the CE/SE Euler solver in [2], in which the corresponding experimental pictures were shown to demonstrate their close

agreements. In reality, the wedge surface may sometimes be covered with dust. The shock wave will interact with the dust while it is reflected from the wedge. This phenomenon involving more complex wave patterns has been observed in the experiments performed by Suzuki[3]. To demonstrate the ability of the CE/SE method to accurately solve some practical physical problems in a simple manner, the reflection of shock waves from a wedge with multi-gutter(used to model the dust layer) is simulated using the 2-D CE/SE Euler solver. Numerical results so obtained are closely resemble the schlieren photographs taken in the laboratory.

(3) **Numerical Simulation of Shock Wave Interaction with Canister**, *in preparation for a paper.*

The interaction of shock wave with a complex geometry canister suspended above an elevated rigid surface has been studied in [4] using a finite-element flux-corrected transport scheme based on unstructed grids with adaptive refinement. The objective of our numerical simulation is to investigate loads exerted on the body during the shock diffraction phase. The CE/SE Euler solver is used to solve this problem on a simple grid with the computational domain divided into several parts. This work demonstrates the ability of the CE/SE method to handle complex geometry in a simple way and its compatibility to use different grids in the same computational domain.

For future studies, the technique of multi-block grids will be used in computations involving different types of grid in neighboring blocks with a consistent interface.

(4) **A Two-Dimensional Euler Solver Based on Varied CEs and SEs**, *in preparation for a NASA Technical Memorandum.*

A 2-D Euler solver has been constructed based on nonuniform grids for handling flow problems involving curved surfaces/bodies, in which the reflecting boundary condition imposed on solid surfaces was implemented. Subsonic, transonic, and supersonic flows past bodies, including circular cylinder and NACA 0012 airfoil, have been simulated using the new solver and reflecting boundary condition. Following an assessment of the present and previous CE/SE results, a conclusion will be made on the choice of reflecting boundary conditions and on related topics that need to be further studied in the CE/SE method develop-

ment. Some of the needed challenging topics have been described here.

The salient features of simplicity, generality and accuracy pertaining to the CE/SE method have been exhibited in all works. As far as applications are concerned, we have shown that this method can be used effectively to solve various flow problems governed by nonlinear partial differential equations, especially those problems having discontinuous solutions which can hardly be computed with traditional numerical methods without relying on ad hoc techniques. To make better use of the CE/SE method, more challenging research topics related to practical flow problems of aircraft and aeroengines are going to be tackled in the future study. These topics are critical in the process of developing the CE/SE method to become a robust, simple, and accurate numerical tool that can be efficiently applied in aerospace industry and at research institutions.

References

1 . T. Aki and F. Higashino,"A Numerical Study on Implosion of Polygonally Interacting Shocks and Consecutive Explosion in a Box," AIP Conference Proceeding 208, 1989, p.167.

2 . X.Y. Wang, C.Y. Chow and S.C. Chang, "Numerical Simulations of Flow Caused by Shock-Body Interaction," AIAA Paper 96-2004, New Orleans, LA, June, 1996.

3 . T. Suzuki, T. Adachi, and S. Kobayashi, "An Experimental Analysis on Shock Reflection over the Two-Dimensional Model of a Dust Layer," in the Proceedings of the 17th International Symposium on Shock Waves & Shock Tube, 1989, p.776.

4 . J.D. Baum, E. Loth, and R. Lohner, "Numerical Simulation of Shock Interaction with Complex Geometry Canister," in the Proceedings of the 17th International Symposium on Shock Waves & Shock Tube, 1989, p.909.